PTC Global **Services**

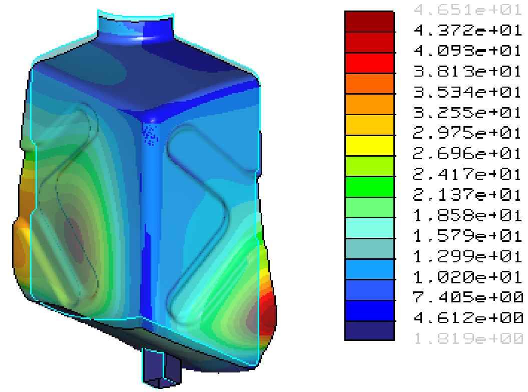
Pro/ENGINEER® Mechanica Wildfire® 4.0 Concept & Contents Workshop Fundamentals I+II Example Cards | Agenda Advanced Workshops

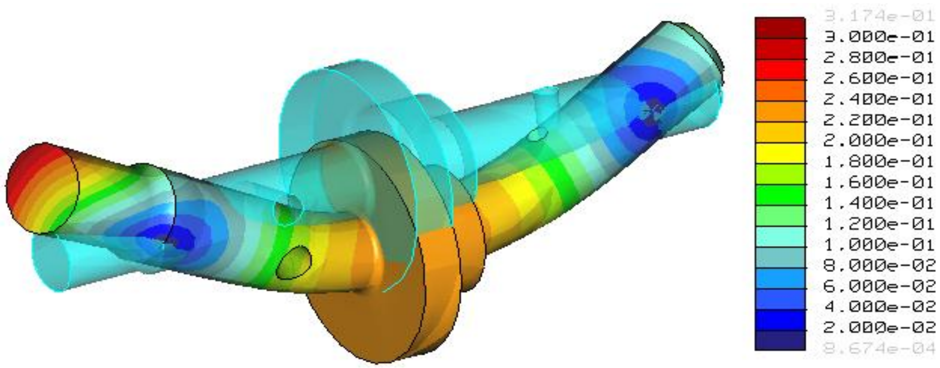
Revision 2.0 | 28-September-2011

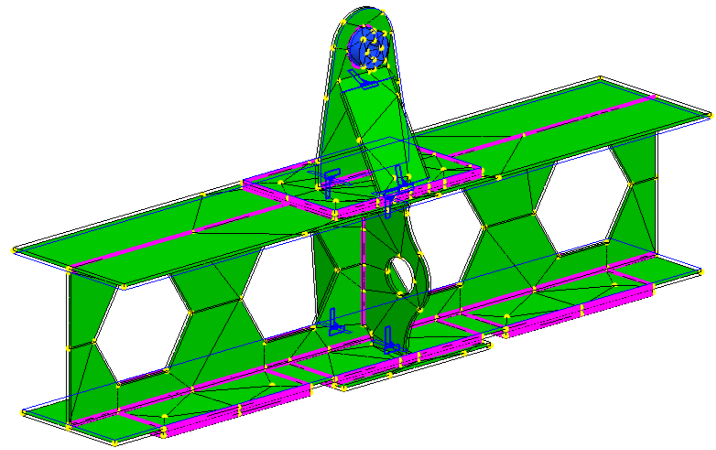
Dr.-Ing. Roland Jakel, PTC CER Simulation Services & Consulting

This document shall make you familiar with the idea and contents of the Fundamentals I+II workshop for Mechanica Wildfire 4. In addition, some card examples from the booklets are shown which are provided in the workshop. Finally, the advanced Mechanica workshops are listed which can be delivered by PTC CER Simulation Services & Consulting.

Contents



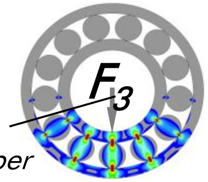




In case of further questions regarding this workshop, comments or corrections, please contact Roland Jakel (rjakel@ptc.com).

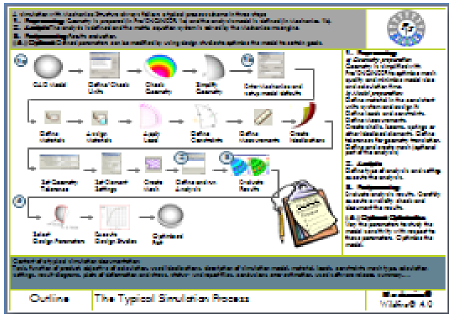
This workshop material contains in a very compacted way theoretical foundations as well as exercises and typical application examples for Mechanica p-finite element simulation. It is provided in a reference card format so that you may use it for your later daily work as an aid to memory.

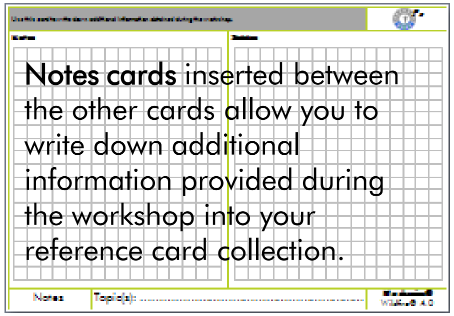
This grey field gives you an overview about the topic treated.



Outline cards

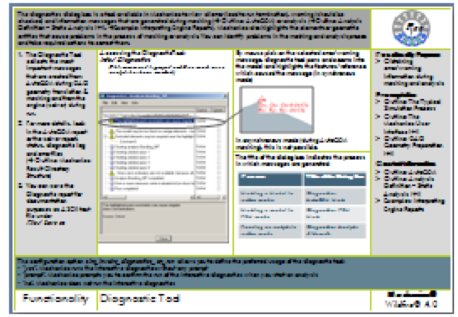
- Introduce you to a certain topic or summarize most important points
- Treat a minimum of theory and give helpful background information

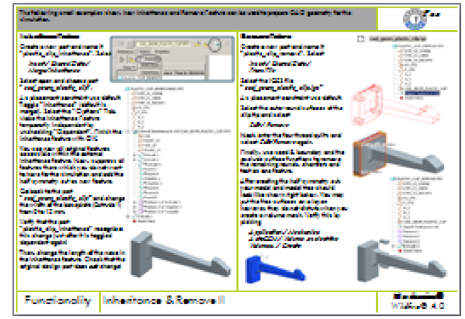




Functionality cards

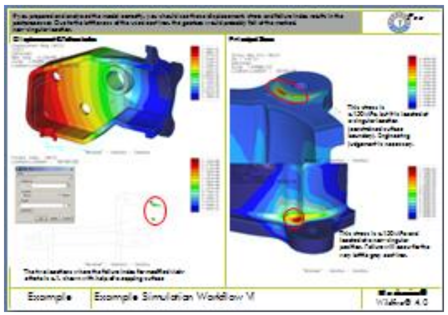
- Introduce to a typical or helpful software functionality
- May, but must not show a small functionality application example





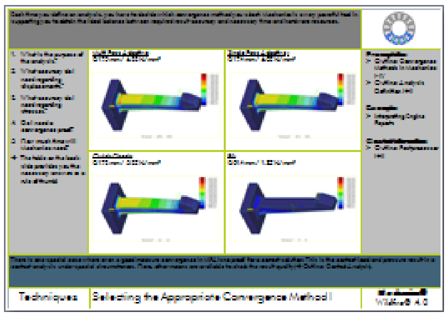
Example cards

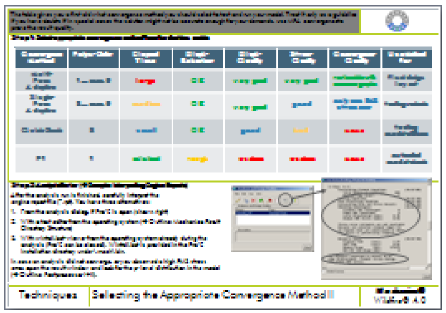
- Guide you through the most important exercise steps, to help you reproducing the more detailed demonstrations given by the instructor
- Show the analysis results to verify you did your analysis steps correctly
- Help you later to apply your knowledge stepwise to your own similar tasks



Techniques cards

- Contain recommended techniques or typical process guidelines to solve certain tasks
- May, but must not show a small exercise to apply the procedure
- Are a reference if you want to apply these techniques later

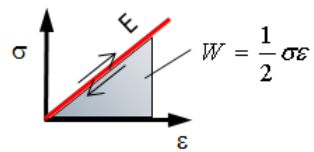




Additional Information, reference literature, tips and tricks may be written in this blue field.

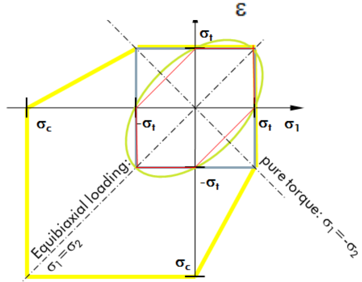
Day 1

Outline: The Workshop Material (F2)
Agenda: Mechanica Workshop – Fundamentals I (F3-4)



1. Introduction into Mechanica FEM simulation

Outline: Mechanica Functionality at a Glance (F5)
Outline: The Finite Element Method (F6)
Outline: The Typical Simulation Process (F7)
Outline: The Mechanica User Interface I+II (F9-10)
Functionality: Diagnostic Tool (F11)
Example: Simulation Workflow I-VI (F13-18)

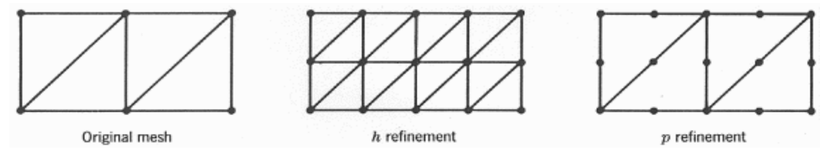


2. Theoretical Foundations

Outline: The h- and p-Version of Finite Elements (F19)
Outline: The p-Method – a Simple Example (F20)
Outline: Structural Mechanics Basics I+II (F21-22)
Outline: Failure Criteria for Linear-Elastic Material (F23)
Outline: Fundamental Equation Systems Solved in Mechanica (F24)

3. Model Preparation

Outline: Mechanica Simulation Application Checklist (F25)
Outline: Units I+II (F27-28)
Outline: CAD Model Preparation I+II (F29-F30)
Techniques: Preparing CAD Geometry for the Analysis I+II (F31-32)
Functionality: Inheritance and Remove I+II (F33-34)

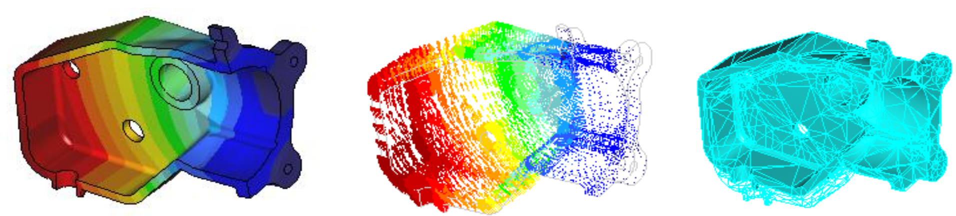


4. Analysis Definition - Basics

Outline: Convergence Methods in Mechanica I-IV: (F35-38)
-Multi Pass
-Single Pass
-Quick Check
-Multi Pass with p=1
Outline: Analysis Definition – Linear Static Analysis I+II (F39-40)
Techniques: Selecting the Appropriate Convergence Method I+II (F41-42)
Example: Interpreting Engine Reports & Using Batch Mode I+II (F43-44)

5. Results Evaluation

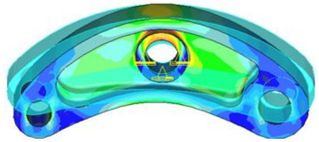
Outline: Postprocessor I+II (F45-46)
Outline: Result Export (F47)
Outline: Mechanica Result Directory Structure (F48)
Example: Evaluating and Exporting Results I+II (F49-50)
Techniques: Assuring Result Quality I+II (F51-52)



The "Mechanica Fundamentals I" Workshop gives you in three days an overview about the basics of Mechanica p-Finite Element Analysis. During the first day, you will refresh theoretic fundamentals; learn techniques to prepare your model for the analysis, learn to select the best suitable convergence method for your actual problem, how to interpret engine reports and how to evaluate results for assuring the result quality.

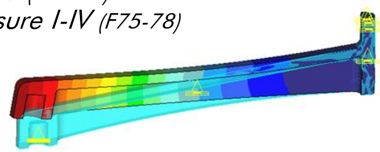
Day 2

1. Materials, Loads and Constraints

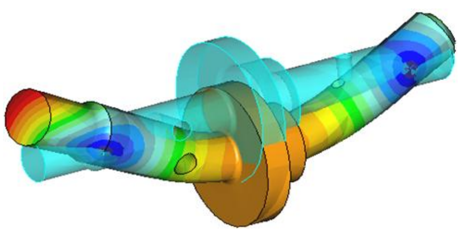


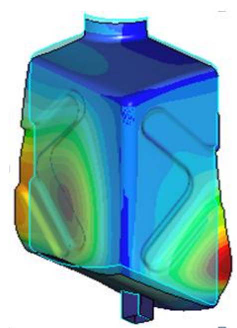
Outline: Materials (F55)
Outline: Mechanica Model Geometry (F56)
Outline: Loads I-III (F57-59)
Outline: Constraints (F60)
Example: A simple tilt lever I+II (F61-62)
Technique: Analyzing a Model in Force Equilibrium I+II (F63-64)
Example: An Office Chair Leg I+II (F65-66)
Technique: Applying a Moment-Free Enforced Displacement I-IV (F67-70)
Example: A Shaft under Bending and Torque I-IV (F71-74)
Example: A Water Tank under Hydrostatic Pressure I-IV (F75-78)

2. Interfaces & Assemblies



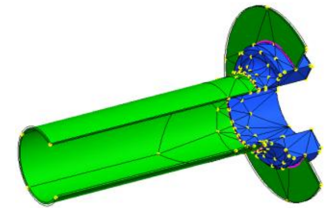
Functionality: Interfaces I+II (F79-80)
Example: A Welded Joint under Global Temperature Load I+II (F81-82)





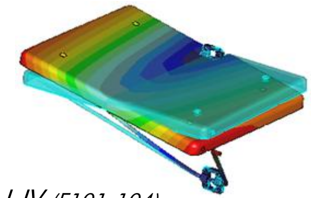
Day 3

1. Meshing

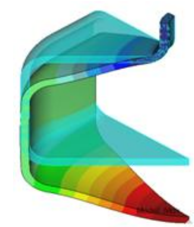


Outline: Meshing/AutoGEM I+II (F83-84)
Example: Meshing a Part I+II (F85-86)

2. Analysis Types



Outline: Large Deformation Analysis (LDA) (F87)
Example: A Thin-Walled Pressurized Vessel I-IV (F89-92)
Outline: Linear Buckling Analysis (F93)
Example: Buckling of a Compression Strut I-IV (F95-98)
Outline: Static Analysis with Prestress (F99)
Example: A Prestressed & Pressurized Sheet Metal Plate I-IV (F101-104)
Outline: Modal Analysis (F105)
Example: Modal Analysis of a Folding Table Assembly I-VI (F107-112)
Outline: Modal Analysis with Prestress (F113)
Example: Modal Analysis of a Violin A String I-IV (F115-118)
Outline: Dynamic Analysis (F119)

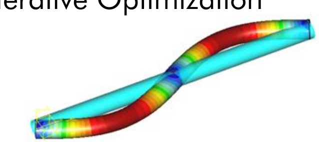


3. Measures

Functionality: Measures (F121)
Example: Measures at a Statically Loaded Lever I-IV (F123-126)

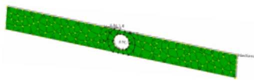
4. Applying Mechanica to Design Tasks: Iterative Optimization

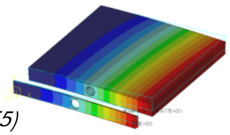
Example: Journeyman's piece I+II (F127-128)



Day 2 will provide you with the most important methods to define loads and constraints, and to define Interfaces for assembly analysis. After day 3, you will be able to mesh your structure with advanced controls, be familiar with the different static strength, stability and modal analyses and know how to use measures. At the end of the Fundamentals I workshop, you will apply your knowledge to iteratively optimize a lever design for minimizing its mass.

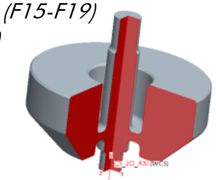
Day 1

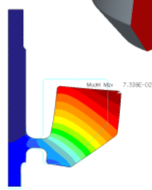




1. Model Types

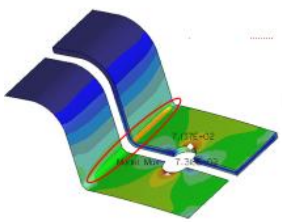
Outline: Model Types – 2D Plane Stress and 2D Plane Strain (F5)
Example: A Bending Beam/Plate in 2D Plane Stress I-IV (F7-F10)
Example: A Bending Beam/Plate in 2D Plane Strain I-IV (F11-F14)
Example: A Bending Beam/Plate as 3D Plane Strain Model I-IV (F15-F19)
Outline: Model Types – 2D Axial and 3D Cyclic Symmetry (F21)
Example: A Flywheel in 2D Axial Symmetry I-IV (F23-F26)
Example: A Flywheel in 3D Cyclic Symmetry I-IV (F27-F30)

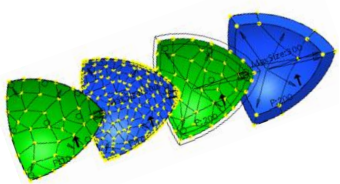


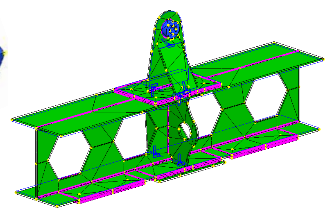


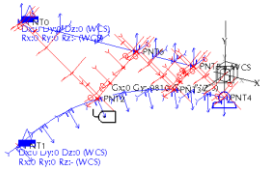
2. Idealizations & Predefined Assembly Connections

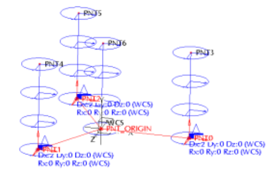
Outline: Shells I+II (F31-F32)
Functionality: Shells on Quilts or Volume Surfaces (F33)
Functionality: Shell Pairs/Midsurfaces (F34)
Techniques: Notches in Shells vs. Notches in Volumes I-III (F35-F37)
Example: A Thin and Thick Pressurized Spherical Vessel I-IV (F39-F42)
Example: Defining Shell Pairs Manually I-III (F43-F45)
Outline: Connection Tools to Join Shell Midsurface Assemblies I+II (F47-48)
Example: A Welded Cantilever with Shells I-IV (F49-F52)



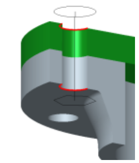






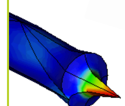


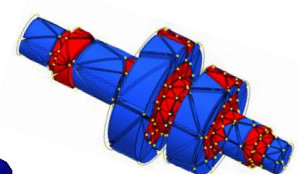
Functionality: Beams I+II (F55-56)
Functionality: Discrete Masses (F57)
Example: A Lattice Truss with Beam Elements I-VI (F59-F64)
Functionality: Rigid Links (F65)
Functionality: Weighted Links (F66)
Example: Functionality Demonstrator for Rigid/Weighted Links I-III (F67-F69)
Functionality: Springs (F71)
Functionality: Fasteners I-IV (F73-76)
Example: Simple Fasteners Connecting a Flange I-IV (F77-80)

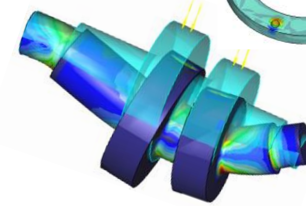


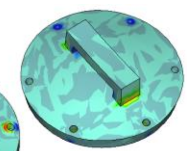
3. Dealing with Singularities

Outline: Singularities I+II (F83-F84)
Example: Singularities at a Friction Gear Shaft I-VII (F85-F91)









Day 1 of the Fundamentals II workshop will deepen your simulation knowledge in more advanced simulation methods, like 2D model types, different idealizations like shells or beams, and predefined assembly connections provided in Mechanica. You will also learn how to mask singularities.

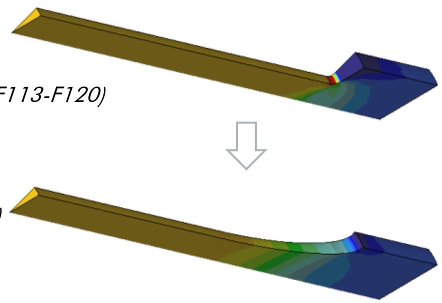
Day 2

1. Sensitivity and Optimization

   Outline: Design Studies I-IV (F93-96)
   Example: Shape Optimization of a Notched Shaft Shoulder (F97-F104)

2. Fundamentals in Contact Analysis

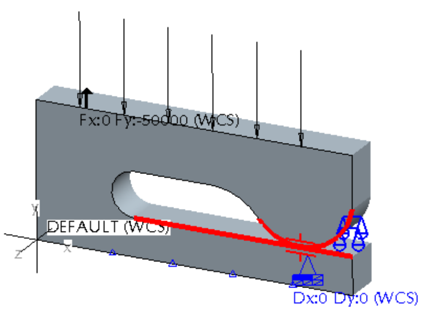
   Overview: Contact Analysis I-VI (F107-112)
   Example: Hertzian Contact at a Clamp I-VIII (F113-F120)

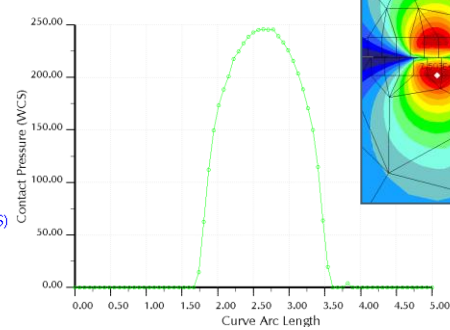


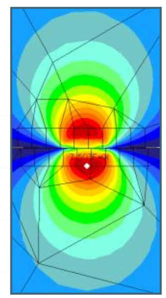
3. Fatal Error – and now?

   Techniques: Model Debugging I+II (F123-F124)

4. Optional: Customer Example Tasks

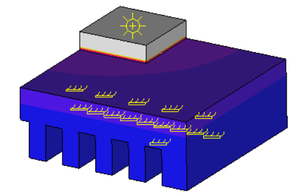


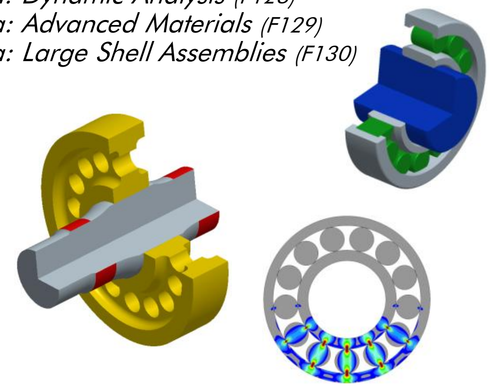




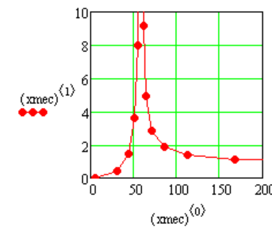
5. Outlook: Advanced PTC CER Simulation Services & Consulting Workshops to deepen your knowledge in applying Mechanica

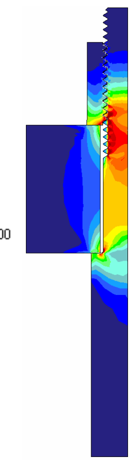
   Agenda: Thermal Analysis (F125)
   Agenda: Contact Analysis (F126)
   Agenda: Fastener Analysis (F127)
   Agenda: Dynamic Analysis (F128)
   Agenda: Advanced Materials (F129)
   Agenda: Large Shell Assemblies (F130)

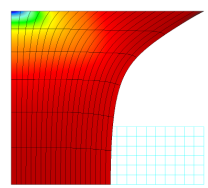


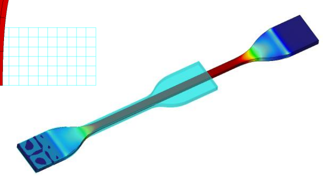


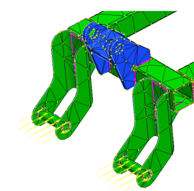
$$x(f) := \frac{e}{\left(\frac{f_0}{f}\right)^2 - 1}$$





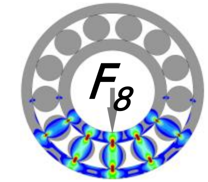






Finally, on day 2 we will treat the Mechanica optimizer, basics in nonlinear contact analysis, and give you aid for debugging your model.

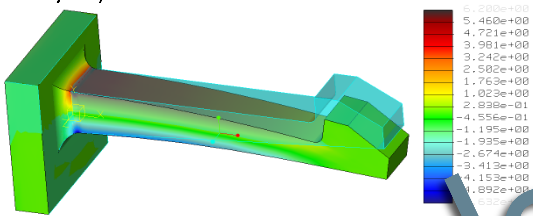
Pro/ENGINEER Mechanica is a structural and thermal analysis code working exclusively with the p-Finite-Element-Method. This assures that the numerical quality of the results delivered can be exactly controlled by the user. It is integrated completely under the Pro/ENGINEER user interface (“Integrated Mode”), even though it can work with an own, classical UI: “Independent” or “Standalone Mode”, which requires a special license. In “FEM Mode”, NASTRAN and ANSYS h-meshes can be prepared for analysis by those FEA solvers and their results postprocessed.



Analysis types supported by Mechanica Structure:

a) Static:

- Linear static analysis (SDA-<u>s</u>mall <u>d</u>isplacement <u>a</u>nalysis)
- Linear static analysis with prestress (based on SDA)
- Static <u>l</u>arge <u>d</u>isplacement <u>a</u>nalysis (LDA) (=external loads are iteratively applied at the deformed structure)
- Static contact analysis (based on SDA)
- Linear Buckling Analysis (stability analysis)



b) Dynamic:

- Modal analysis
- Modal analysis with prestress
- Dynamic frequency analysis
- Dynamic time analysis
- Random response analysis
- Dynamic shock analysis (“earthquake” analysis)

Analysis types supported by Mechanica Thermal:

- Steady state thermal analysis
- Transient thermal analysis

Remarks:

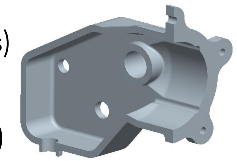
Mechanica Thermal allows to analyze temperature fields and heat flows <u>within the mechanical structure</u> for given thermal boundary conditions and heat loads. It is no CFD tool which estimates these conditions automatically!

Temperature fields analyzed by Mechanica Thermal can be read into Mechanica Structure to calculate thermal displacements and stresses.

Additional **Design Studies** allow to study the influence of design variables to the model and optimize it to certain goals.

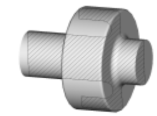
Model types supported:

- 3D volumes (element types: tetrahedrons, wedges, bricks)
- 2D plane stress (element types: triangles, quadrilaterals)
- 2D plane strain (element types: tris, quads, 2D shells)
- 2D axial symmetric (element types: tris, quads, 2D shells)



Idealized element types supported:

- p-shells (element types: tris, quads)
- p-beams
- Rigid links
- Weighted links
- Discrete masses
- Discrete springs



Predefined assembly connections supported:

- Features to connect shell midsurfaces in assemblies:
  Welds (manually defined shell elements)
  Bonding elements (automatically created orthotropic shells/volumes)
  Assembly links (automatically created multi point constraints - MPCs)
- Spot welds (“beam fasteners”)
- Fasteners (idealized bolts)

Materials supported:

- Linear elastic:
  a) Isotropic (direction independent properties)
  b) Orthotropic (direction dependent properties)
  c) Composites (for shells only)
- Hyperelastic (for example elastomer, rubber)
- Plasticity is supported in Wildfire 5.0

Because of the integration in the Pro/ENGINEER CAD software, during preprocessing Mechanica works directly on the CAD geometry. This makes Mechanica very powerful when changes or iterative design optimizations are performed, since the simulation features are associatively linked to the CAD geometry. Just before the meshing starts the CAD geometry is translated into the simulation geometry on which the Finite Element analysis is based. Results can be evaluated with an own post processor providing extensive functionality. Both simple parts up to huge and complex assemblies can be analyzed with Mechanica!

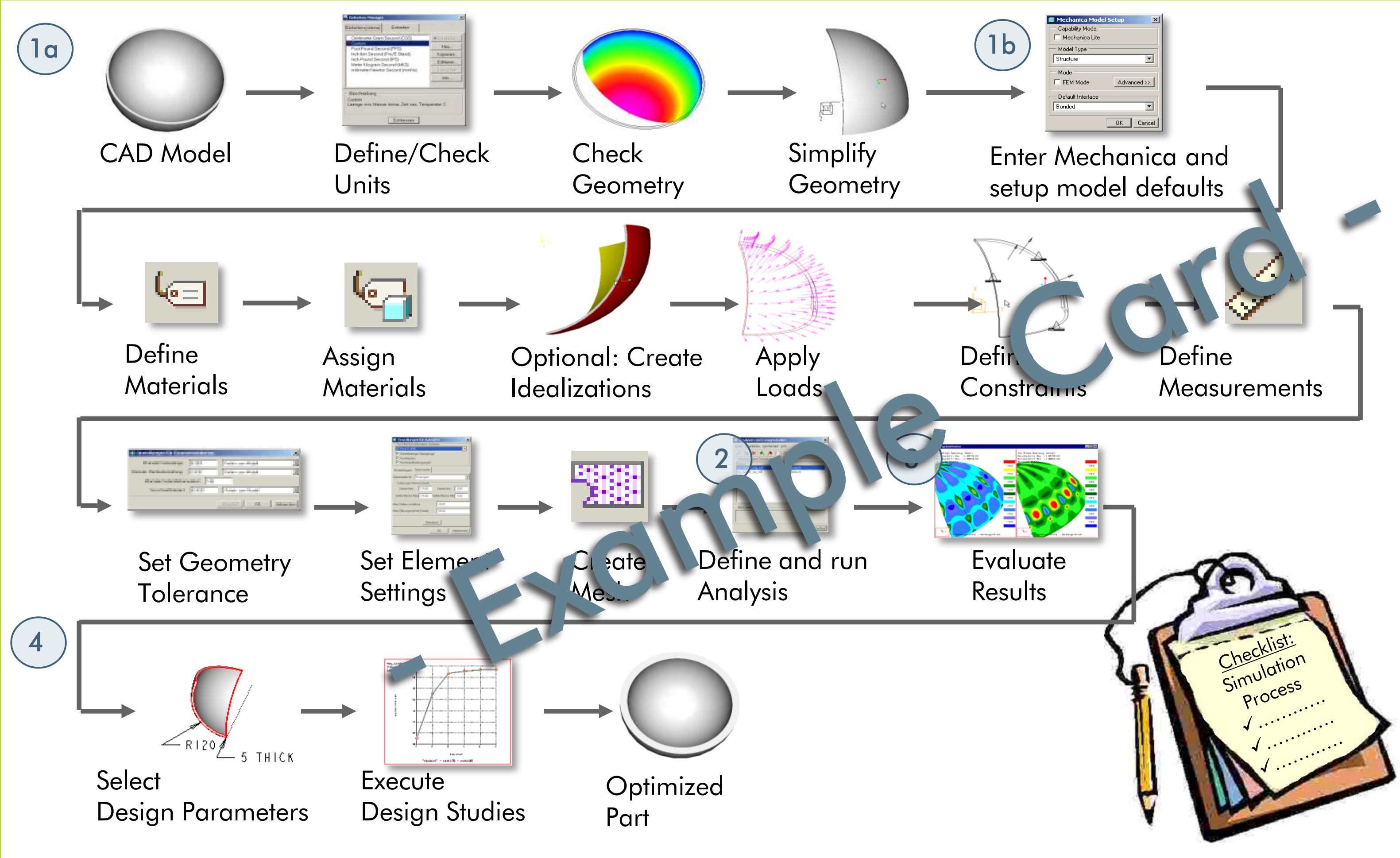
A simulation with Mechanica Structure always follows a typical process scheme in three steps with an optional fourth:

1. Preprocessing: Geometry is prepared (in Pro/ENGINEER, 1a) and the analysis model is defined (in Mechanica, 1b).

2. Analysis: The analysis is defined and the matrix equation system is solved by the Mechanica msengine.

3. Postprocessing: Results evaluation.

(4.) Optional: Defined parameters can be modified by using design studies to optimize the model to certain goals.



1. Preprocessing:

a) Geometry preparation

Geometry is simplified with Pro/ENGINEER to optimize mesh quality and minimize model size and calculation time.

b) Model preparation:

Define material in the consistent units system and assign it.
Optional: Create shells, beams, springs or other idealized elements.
Define loads and constraints.
Define Measurements.
Define tolerances for geometry translation.
Define and create mesh (optional part of the analysis).

2. Analysis:

Define type of analysis and settings, execute the analysis.

3. Postprocessing:

Evaluate analysis results. Carefully execute a validity check and document the results.

(4.) Optional: Optimization

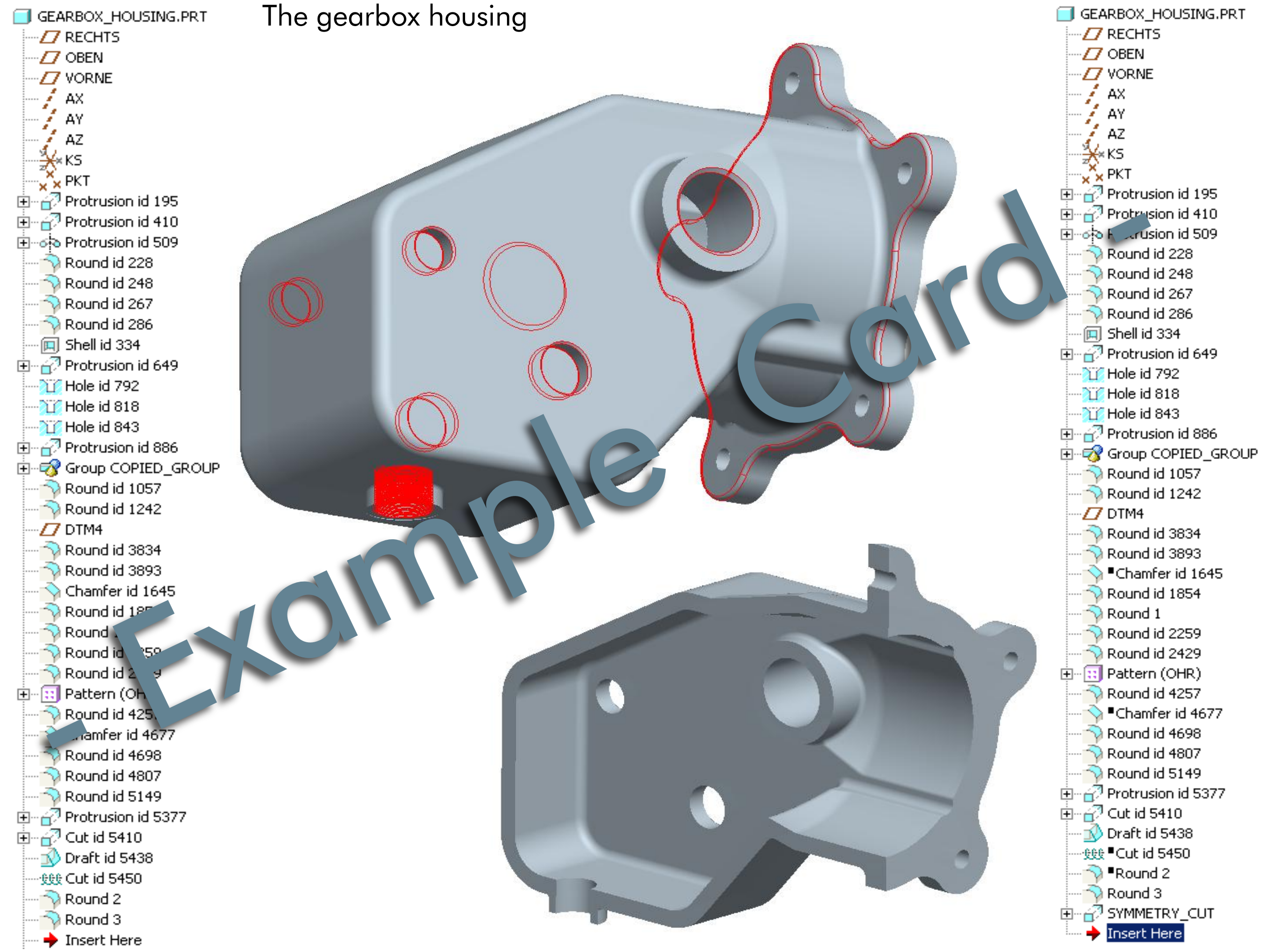
Vary the parameters to study the model sensitivity with respect to these parameters. Optimize the model.

Content of a typical simulation documentation:
Task, function of product, objective of the simulation, geometry simplification, used idealizations, description of the simulation model, material, loads, constraints, mesh type, calculation settings, result diagrams, plots of deformation and stress, status- and report files, conclusions, error estimation, used software release, summary.

This part shows a special gearbox housing made of gray cast iron. The reaction forces of the shafts are known from a separate gear analysis, from this analysis we conclude that the housing can be analyzed half-symmetric. We are now interested in if the brittle cast housing can withstand these bearing reaction loads.

1. Open the part "*gearbox_housing*". Check the units system.
2. Examine the CAD geometry and search for features to be suppressed for the simulation.
3. Identify and suppress these features.
4. Cut half of the housing away for symmetry reasons.

The gearbox housing



Exercise Purpose:
- ➢ Get known with the typical workflow necessary for a simple linear static analysis in Mechanica
- ➢ Get known with the Mechanica UI

Prerequisites:
- ➢ Outline: The Typical Simulation Process
- ➢ Outline: The Mechanica User Interface

Context Information:
- ➢ Functionality: Diagnostic Tool
- ➢ Techniques: Preparing CAD Geometry for the Analysis
- ➢ Outline: Units

This example shows a simple, but complete simulation process for a single part under quasi-static load. Keep in mind the simulation process as guideline (→Outline: The Typical Simulation Process).

Define material and apply bearing loads:

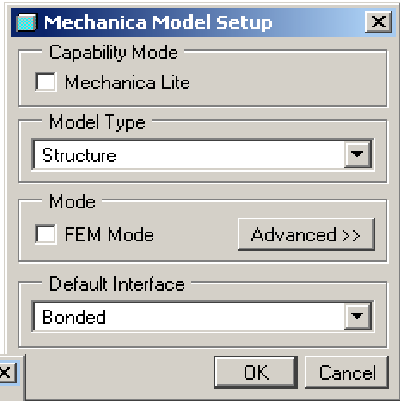
F_{11}

Step 5:

Enter Mechanica:

Application / Mechanica

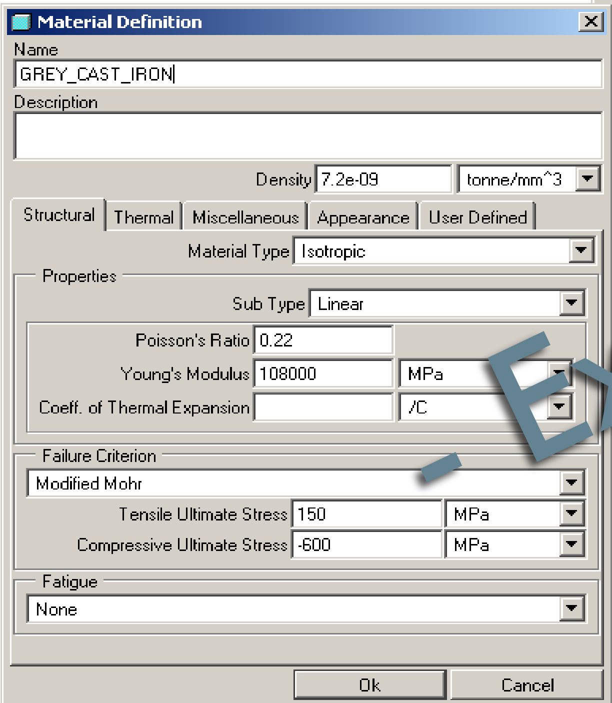
Check the Model Setup settings.



Define Material:

Properties / Materials… / File / New

Enter the material data shown.



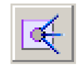
Assign the Material to the part:

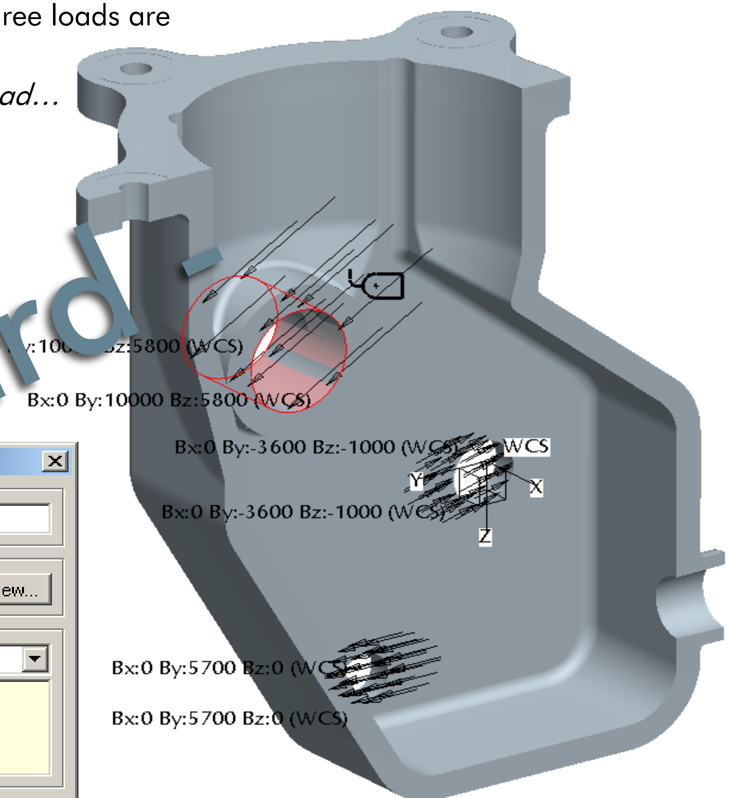
Properties / Material Assignment…

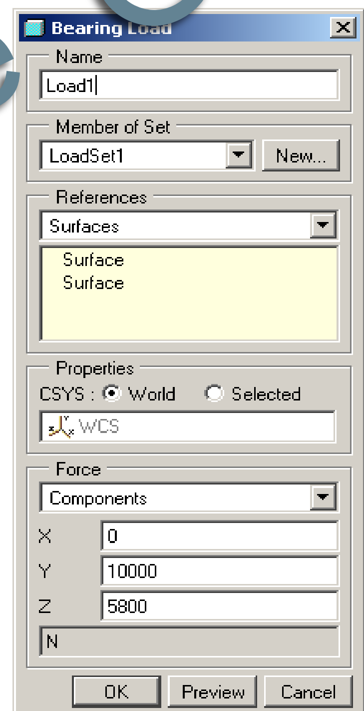
Step 6:

Apply the loads. All three loads are bearing loads:

Insert / Bearing Load…

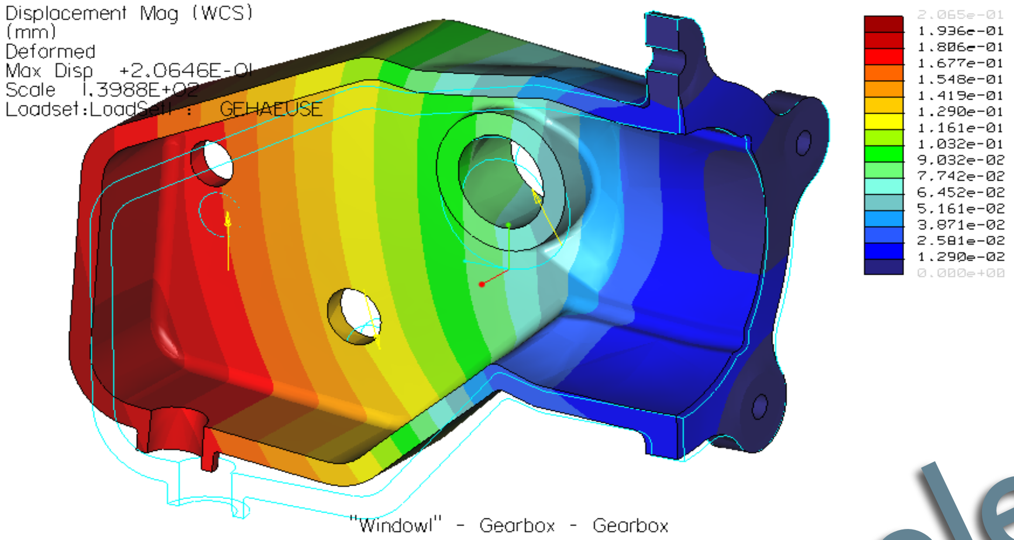


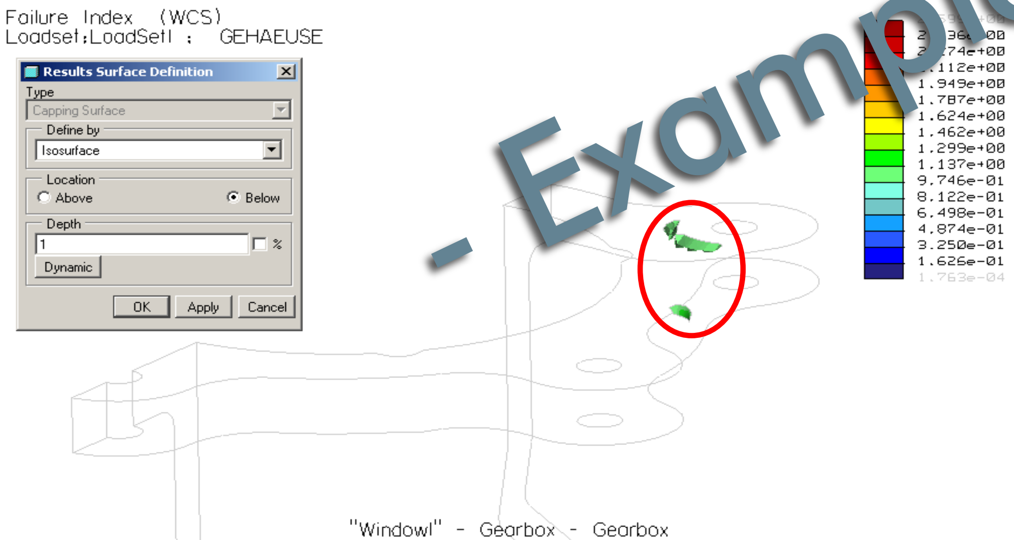




If you prepared and analyzed the model correctly, you should see these displacement, stress and failure index results in the postprocessor. Due to the brittleness of the used cast iron, the gearbox would probably fail at the marked, non-singular location.

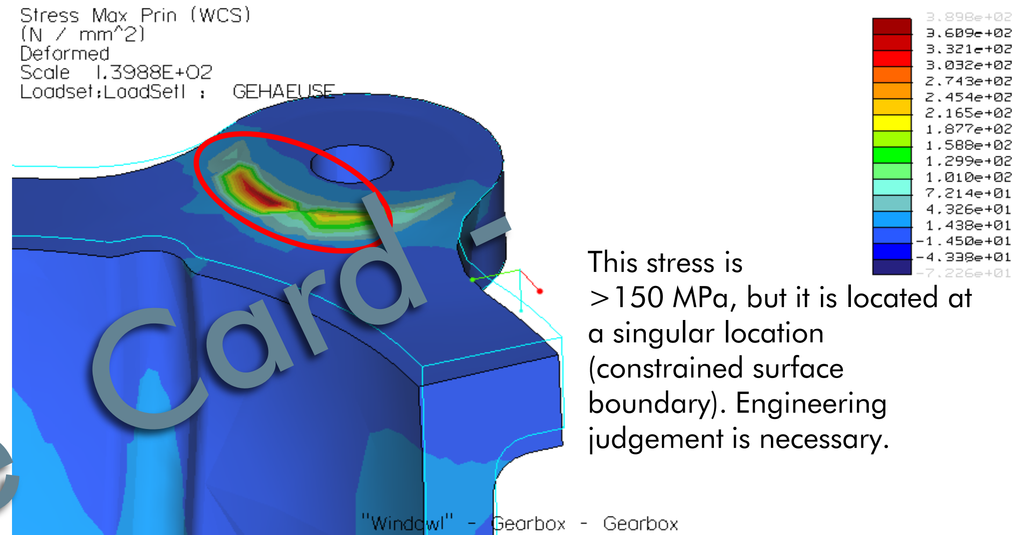
Displacement & Failure Index



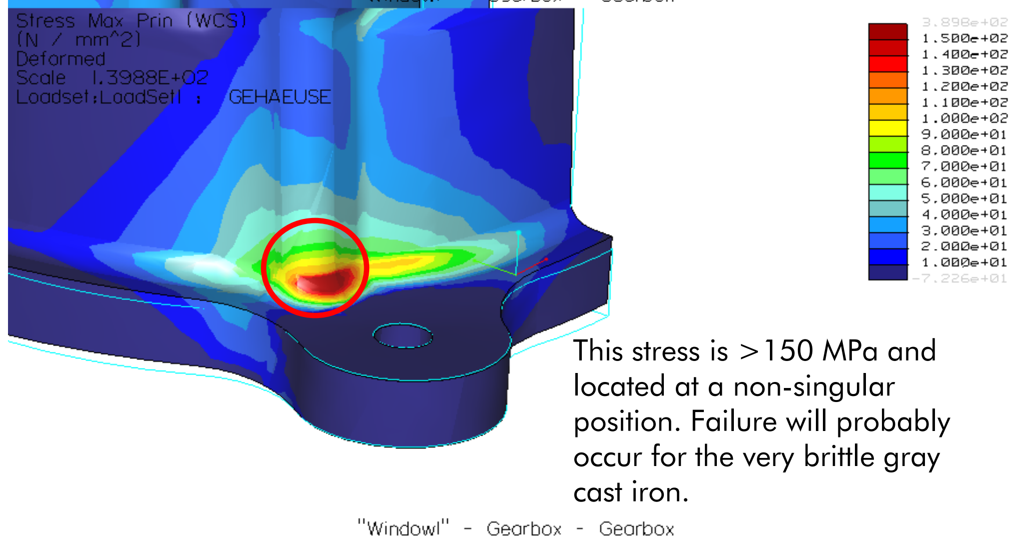


The two locations where the failure index for modified Mohr criteria is >1, shown with help of a capping surface

Principal Stress



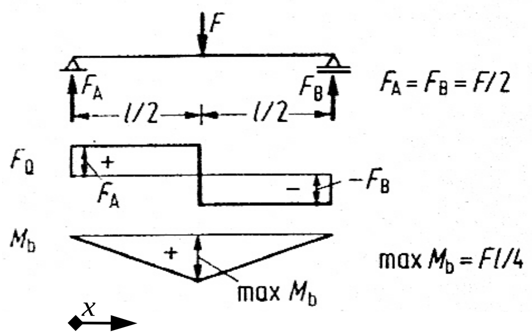
This stress is >150 MPa, but it is located at a singular location (constrained surface boundary). Engineering judgement is necessary.



This stress is >150 MPa and located at a non-singular position. Failure will probably occur for the very brittle gray cast iron.

A basic understanding of the p-method is necessary, so that the user can define the model and analysis correctly to take full advantage of the p-method's benefits. This very simple example shall help to understand how it works. Look at the deformed shape and stress distribution as well as the maximum deflection and longitudinal stress values when the p-level of the single p-element used increases!

Example: Simply supported 3-point bending bar:



$$M_{b\max} = \frac{Fl}{4}$$

$$\sigma_{b\max} = \frac{M_{b\max}}{w_b}; \quad w_b = \frac{bh^2}{6}$$

$$f_{\max} = \frac{Fl^3}{48EI_y}; \quad I_y = \frac{bh^3}{12}$$

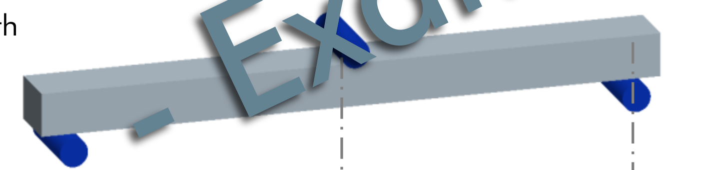
$$w(x) = \frac{Fl^3}{48EI_y}\left[3\frac{x}{l} - 4\left(\frac{x}{l}\right)^3\right]$$

We have a simple bending bar with loaded length l=40 mm, width b=4 mm, height h=3 mm. Material is steel with E=200 GPa. We apply a force of 200 N. From the analytical solution, we obtain:

$$M_{b\max} = 2000\,Nmm; \quad w_b = 6\,mm^3; \quad I_y = 9\,mm^4$$

$$\sigma_{b\max} = 333.3\,MPa; \quad f_{\max} = 0.1481\,mm$$

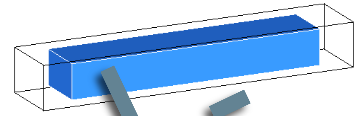
The CAD-geometry with the load rollers looks as follows:



We use half symmetry in the idealized physical model and cut away the load-free beam end:

Solution with the p-method in Mechanica:

We use just one p-brick element to mesh the half beam. 1% convergence is requested on measures: displacements, strain energy, max. and min principal stress, von Mises and longitudinal stress.



<u>The results are:</u>

p=1:
0.0082[illegible]6 mm
10.78 MPa

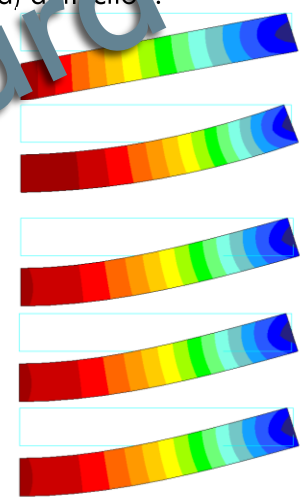
p=2:
0.1190 mm
191.79 MPa

p=3:
0.1507 mm
334.52 MPa

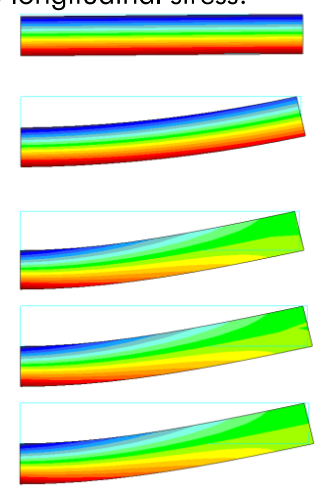
p=4:
0.1507 mm
334.57 MPa

p=5:
0.1507 mm
335.23 MPa

a) deflection:



b) longitudinal stress:



<u>Remarks:</u>

- The max. deflection in the displacement plot is scaled to 20 % of the model size, in the stress plot to 20 times the absolute value
- Plotting grid in the plots was set to 10!
- Equations from the simple beam theory on the left side do not take into account additional deflection from shear stresses!

In this example of the simple bending bar, we know from the analytical solution that the deflection curve w(x) is a polynomial function where the x-coordinate (length) appears as third potency. Hence, adding higher order functions with p>3 does not significantly change the solution any more!

Note: Across the brick element, there is some linear variation of stress even at p=1. Consider a quad: The displacement is linear in x and y, so u = (a+bx)*(c+dy). du/dx is linear in y, while du/dy is linear in x. In tetrahedrons or tris, stress at p=1 is really constant!

The RMS Stress Error Estimate is a local error norm, which is provided for all Mechanica convergence methods. Stress error estimates are not available just if all external element edges lie in regions that are constrained or where the stress is potentially singular.

RMS Stress Error Estimate:

This really is a local stress error estimate and should not be mixed up with the Global RMS Stress Index, which refers to global strain energy!

In both SPA and MPA, it is the maximum over all edges in the model of the RMS stress error along one edge. Stress error is estimated at sample points along an edge by comparing smoothed ("superconverged") and non-smoothed stresses. The stress error excludes regions of potential singularities (constraints, reentrant corners)!

RMS Stress Error Estimates are computed as follows:

σ = the directly computed stress
(from the derivatives of the displacement polynomials)
σ_m = the smoothed ("superconverged") stress.

mean-square stress error = integral over edge of $(\sigma - \sigma_m)^2$

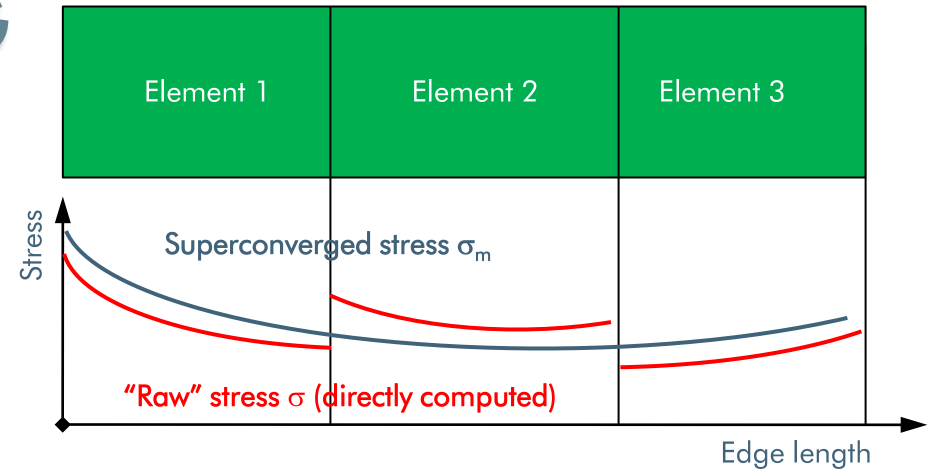
RMS Stress Error = square root of the mean-square stress error

Superconverged (smoothed) stresses:

To represent the smoothed stress, Mechanica uses the same polynomial functions like for the displacements. The coefficients of the stress polynomials are found by least squares fitting to stresses calculated at various sample points within each element.

In this way, the typical "stress jumps" (discontinuous stresses) of the "raw" or directly computed stresses at the element boundaries disappear. It turns out that the smoothed stresses calculated as above often converge faster than non-smoothed stresses.

The term "superconvergence" comes from the now classic paper [1].

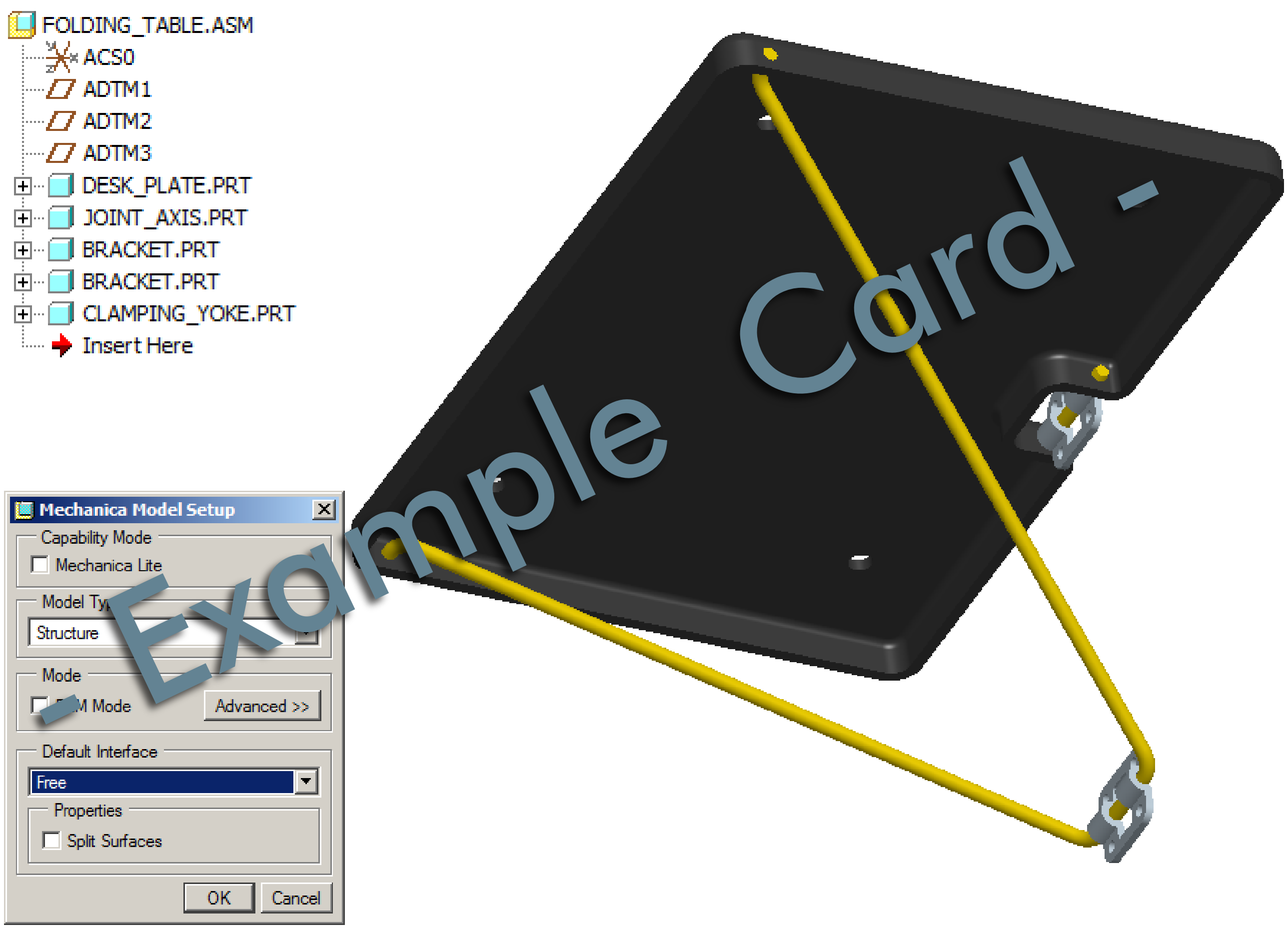


Reference:
[1] Zienkiewicz, O, and Zhu, J, "A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis", Int. J. Numer. Methods Engr. 24 (1987), 337-357.

In this example we want to analyze the fundamental frequencies of a folding table. What makes this analysis tricky is that we have an assembly in which the single parts are not merged together (e.g. glued or welded), but have certain degrees of freedom that allow a relative movement: We have different joint types in the model! We will learn a technique how to deal with that. This technique can be used for static analyses, too. It is a linearization of a contact problem: As a consequence the solution around this linearization may be inaccurate. Anyway, the global results should be fine.

1. Open assembly *Folding Table* and enter Mechanica
2. Set the default interface to *Free*: *Edit / Mechanica Model Setup…* This step simplifies joint definition since all touching surfaces are not merged, but joints!
3. Assign material PVC to the desk plate and the two brackets
4. Assign material SS (stainless steel) to the joint axes and the clamping yoke.
5. Examine all on part level predefined simulation features:
   - volume regions
   - points
   - weighted links

The folding table assembly



Exercise Purpose:
- Understanding modal analysis
- Idealizing joints to analyze assemblies

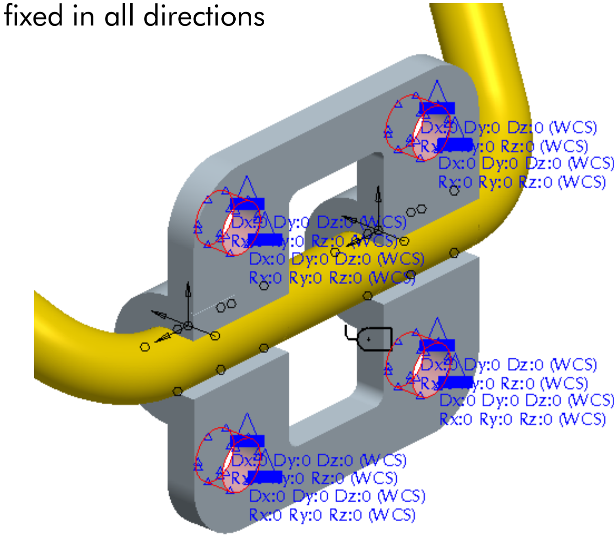
Prerequisites:
- Outline: Modal Analysis
- Functionality: Interfaces

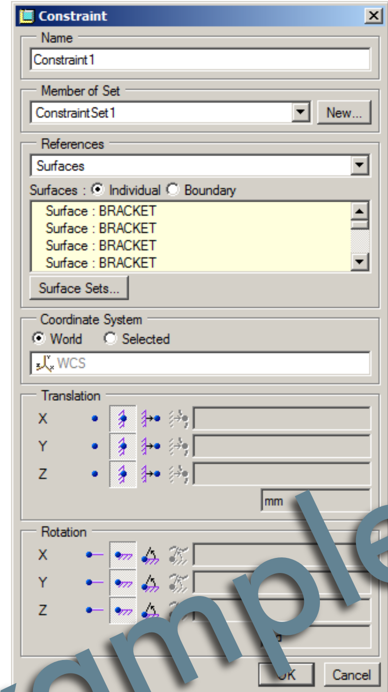
Context Information:
- Functionality: Springs
- Functionality: Rigid and Weighted Links (for both, see the booklet "Workshop Fundamentals II")

It is a good practice to idealize joints in assemblies undergoing a linear analysis with a pair of weighted links and an advanced spring in between their dependent nodes. With setting the appropriate spring tensor elements "infinite" stiff, any possible relative motion can be enabled. Note not to use too high values in the spring tensor to represent infinity; in this example e.g. 1.0E+10 N/mm – not more – is a good choice. Higher values may lead to inaccurate results or even an analysis failure like "the model is insufficiently constrained for the analysis".

6. Define the constraints

All holes of both brackets have to be fixed in all directions

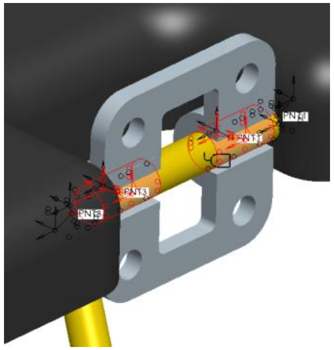


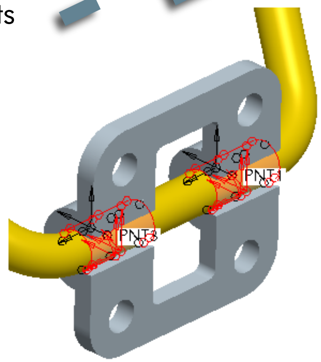


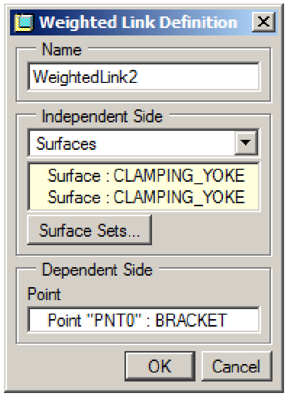
7. Create weighted links

at the following eight locations on assembly level:

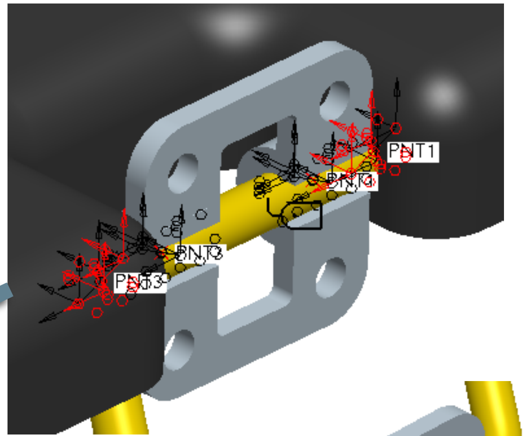
a) At the lower clamping yoke and upper axis within the brackets





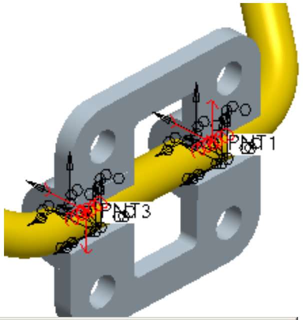


b) At the upper axis within the desk plate holes (Note: Highlighting an existing weighted link and RMB "Info" shows you to which surface it is already connected!)

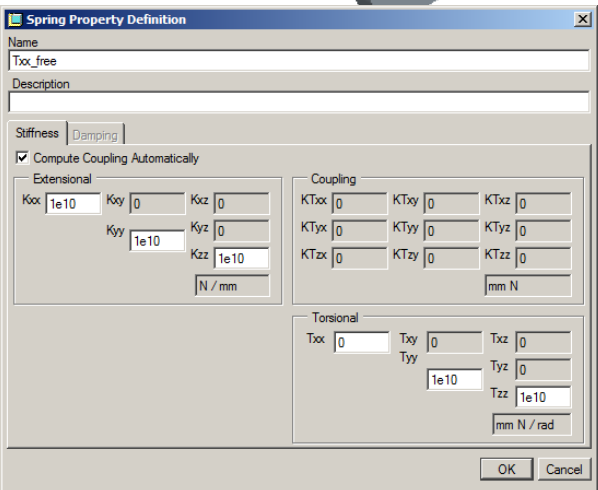


c) At both holes in the desk plate where the clamping yoke ends

8. Create advanced springs between the dependent nodes of the weighted link pairs



a) For the four joint positions at the two brackets, use a tensor where just the rotation around the local spring X-axis is free. For the local spring coordinate system, see →Functionality: Springs.



Note: This linearization is a simplification compared to reality. The bearings also have friction (nonlinearities can not be treated in a modal analysis accurately), and due to manufacturing tolerances, we may have gaps resulting into excitation force dependent fundamental frequencies.

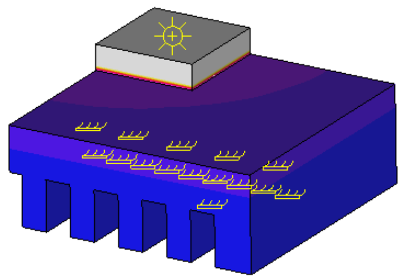
PTC CER Simulation Services & Consulting also offers different advanced workshops (training material is not contained in this booklet). For information only, their possible contents are shown on this card. Training content is subject to change and can be adopted individually to the customer's needs.

Day 1

1. Introduction into Thermal Analysis

Outline: Functionality overview of Mechanica Thermal
- *Functions*
- *Limitations*

Outline: Basics in heat transfer analysis
- *Conductivity*
- *Heat transfer*
- *Heat sources*
- *Unit systems*



2. Steady-State Thermal Analysis

Outline: Analogy heat transfer analysis – static structural analysis
Outline: Thermal loads and boundary conditions
- *Thermal interfaces*
- *Material definition*
- *Analysis definition*

Outline: Postprocessing
Example: Thermal analysis of a computer chip
Example: Thermal analysis of a heat pipe
Example: Thermal analysis of a furnace console

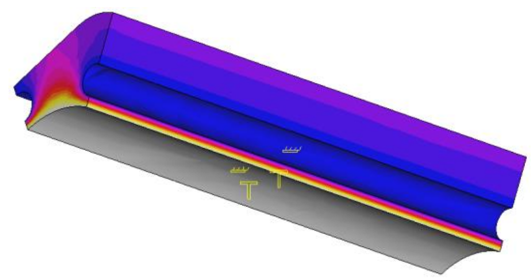
3. Transient Thermal Analysis

Thermal loads and boundary conditions
Material definition
Measures
Analysis definition
Postprocessing
Example: Transient thermal analysis of a bi-metal

4. Thermo-Mechanical Analysis

Outline: Loads definition
- *Global temperature loads*
- *MechT Loads*
- *External loads*

Outline: Material definition
Outline: Analysis definition
Example: Deformation of a bi-metal when it is completely heated up
Example: Deformation of a bi-metal under steady-state thermal load
Example: Deformation of a bi-metal under transient loading



This 1-day workshop guides you through all the functionality of the Mechanica Thermal module.

Agenda Pro/ENGINEER® Mechanica Wildfire® 4.0
Workshop Contact Analysis
PTC CER Simulation Services & Consulting also offers different advanced workshops (training material is not contained in this booklet). For information only, their possible contents are shown on this card. Training content is subject to change and can be adopted individually to the customer's needs.

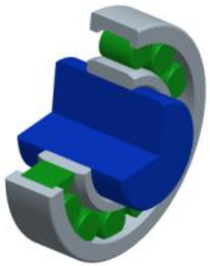
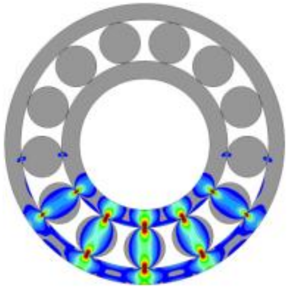
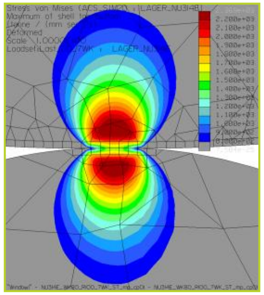
Day 1

1. Introduction into Contact Analysis

Mechanica functionality overview
Theoretical Background:
- *The Penalty Method used in Mechanica for contact analyses*
- *Modified Newton-Raphson for the iteration loop*
- *Obtaining convergence in contact analysis*
- *Mesh control in contact analysis*
- *Assuring result quality with measures and postprocessing*
- *Engine command line options and environment variables for advanced analysis control*

2. Friction-free Contact Analysis

Selecting the best suited model type
Example: A clamp with Hertzian contact under pressure
Example: A shaft-hub connection with shrink-fit as 2D-axial symmetric model
Example: A shaft-hub connection with shrink-fit as 3D segment model
Advanced Example: A fully-detailed cylindrical roller bearing with Hertzian pressure as 2D-plane strain model

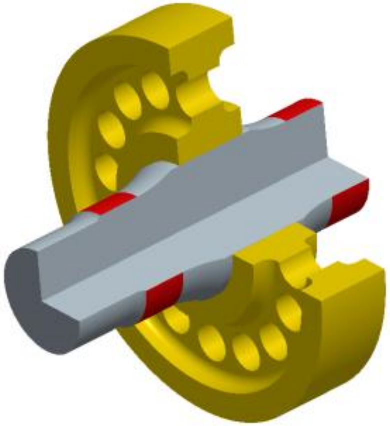
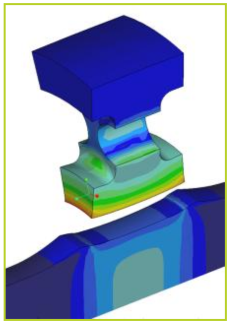
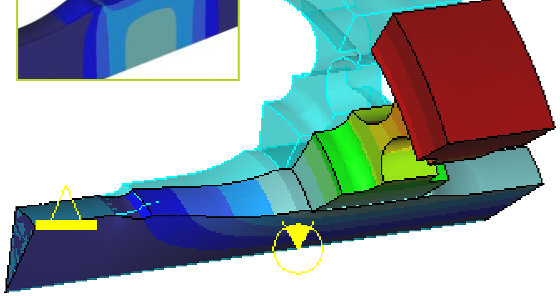




4. Introduction into the Infinite Friction Contact Model

Principle of the infinite friction contact model
Infinite friction specific measures (slippage measures)
Assuring result quality with measures and postprocessing

5. Infinite Friction Contact Analysis

Example: Two bricks in contact under pressure and shear with specific measures
Example: Torque-loaded shaft-hub connection with shrink fit as 3D segment model

Remark: This 1-day workshop is prerequisite for the 2-day-workshop "Fastener Analysis"

Agenda Pro/ENGINEER® Mechanica Wildfire® 4.0
Workshop Fastener Analysis
PTC CER Simulation Services & Consulting also offers different advanced workshops (training material is not contained in this booklet). For information only, their possible contents are shown on this card. Training content is subject to change and can be adopted individually to the customer’s needs.

Day 1

1. Introduction into Fastener Analysis

Outline: Theoretical background
- *Fastener theory & bolt diagram*
- *Characteristic quantities for fastener analysis*

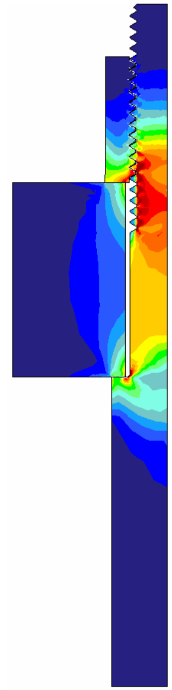
2. The Centrally Loaded Fastener Connection

Example: A fully-detailed bolted piston under hydraulic pressure as 2D axial symmetric model

3. The Mechanica Fastener Feature

Functionality: The fastener feature
- *Theoretical background and application*
- *Limitations*

Example: The bolted piston under hydraulic pressure with the fastener feature with contact
Example: The bolted piston under hydraulic pressure with the fastener feature with linearized substitute contact (“separation stiffness”)
Technique: Reducing the number of necessary analyses when working with the linearized substitute contact
Example: An eccentrically loaded fastener in a flange under tension with the fastener feature with contact
Example: An eccentrically loaded fastener in a flange under tension with the fastener feature with linearized substitute contact (“separation stiffness”)



Day 2

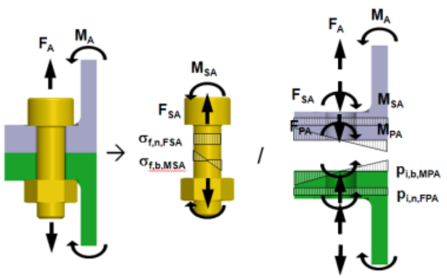
1. Eccentrically Loaded Fasteners

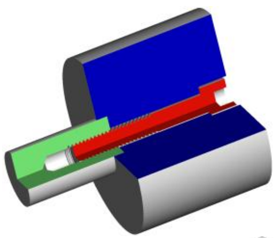
Outline: Theory of loading
Outline: Methods to idealize eccentrically loaded fasteners
Example: An eccentrically loaded fastener in a flange under tension using volumes for the fastener and contact
Example: An eccentrically loaded fastener in a flange under tension with a beam and contact
Example: An eccentrically loaded fastener with linearized substitute contact and specific measures

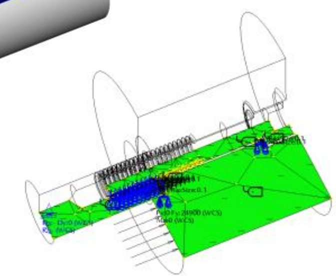
2. Fastener Assembly Analyses

Techniques: Analyzing big assemblies containing many fasteners

3. Customer Examples & Projects







Remark: This course requires as prerequisite the 1-day-workshop “Contact Analysis”

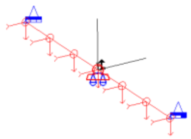
PTC CER Simulation Services & Consulting also offers different advanced workshops (training material is not contained in this booklet). For information only, their possible contents are shown on this card. Training content is subject to change and can be adopted individually to the customer's needs.

Day 1

1. Introduction into Dynamic Analysis

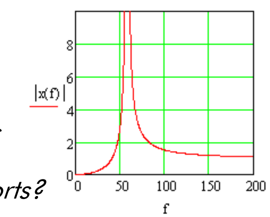
The differential equation Mechanica solves in modal and dynamic analysis
The modal transformation
Mass participation factors
Overview of dynamic analysis types in Mechanica
Measure definitions in dynamic analysis

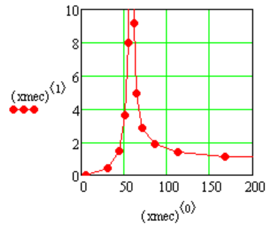
$$x(f) := \frac{e}{\left(\frac{f_0}{f}\right)^2 - 1}$$



2. Advanced Modal Analysis

Definition and options
Modal stresses
Example: A constrained structure
Example: An under-determined (free rotating) shaft
Example: A free-free modal analysis





3. Dynamic Frequency Analysis

Theory of dynamic frequency analysis
Definition and options
Output relative to ground or to supports?
Force and base excitation
Phases
Example: A one-mass oscillator
Example: An unbalanced rotating shaft
Example: An exhaust pipe bolted to a cylinder (base excitation)

4. Dynamic Time Analysis

Theory of dynamic time analysis
Definition and options
Defining forcing functions
Example: A one-mass oscillator
Example: An unbalanced rotating shaft – transient and steady state
Example: A part subjected to an half sine shock

Day 2

1. Random Response Analysis

Theory of random response analysis
Definition and options
Defining acceleration spectral density, measures
Example: A one-mass oscillator
Example: A circuit board under random vibration

2. Dynamic Shock Analysis ("Earthquake Analysis")

Theory of dynamic shock analysis
Definition and options
Modal superposition methods
Example: A one-mass oscillator
Example: A cubicle of electronic equipment subjected to an earthquake

Depending on customer demand, the workshop can be reduced to 1-day duration if not all dynamic analysis types are of interest.

Workshop Advanced Materials

PTC CER Simulation Services & Consulting also offers different advanced workshops (training material is not contained in this booklet). For information only, their possible contents are shown on this card. Training content is subject to change and can be adopted individually to the customer's needs.

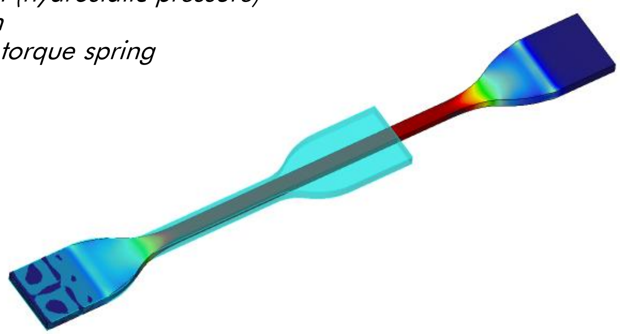
Day 1

1. Introduction into Hyperelasticity

Review of Hooke's law for linear elastic materials
The specific strain energy of linear elastic materials
Behavior of hyperelastic materials
Material laws for hyperelastic materials
About selecting the material model and performing tests
Implementation of hyperelastic material laws in Mechanica
Defining hyperelastic material parameters in Mechanica
Test set-ups and specimen shapes of the supported material tests
The uniaxial compression test
Stress and strain definitions in the Mechanica LDA analysis

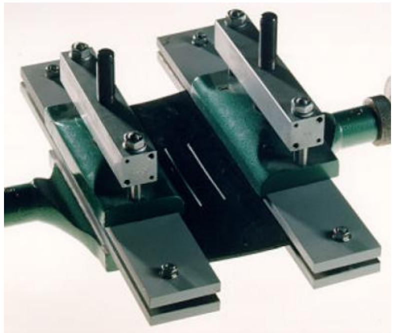
2. Application Examples with Hyperelastic Material

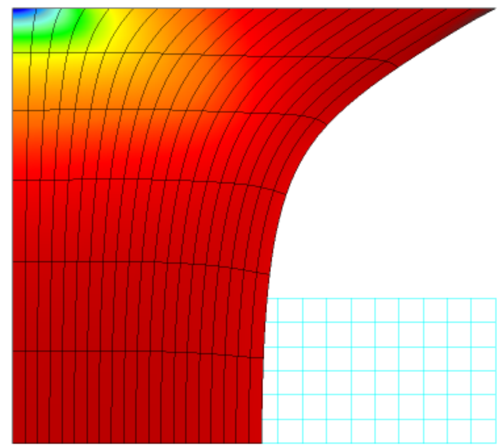
Example: Uniaxial tension test
Example: Equibiaxial tension test
Example: Volumetric test (hydrostatic pressure)
Example: Planar tension
Example: An elastomer torque spring



3. Optional: Orthotropic Material & Composites

4. Optional: Plasticity (starting with Wildfire 5)





Unlike in Mechanica Wildfire 4, in Creo Simulate it is possible to combine the different material and geometric nonlinearities (LDA, contact analysis, hyperelastic & elasto-plastic material) in a static analysis.

Agenda Pro/ENGINEER® Mechanica Wildfire® 4.0
Workshop Strategies for Large Shell (Welded) Assemblies and Analyses
PTC CER Simulation Services & Consulting also offers different advanced workshops (training material is not contained in this booklet). For information only, their possible contents are shown on this card. Training content is subject to change and can be adopted individually to the customer's needs.

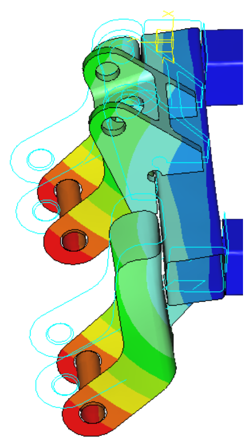
Day 1

1. Introduction

Shell or volume model?
When to use shell models
When not to use shell models

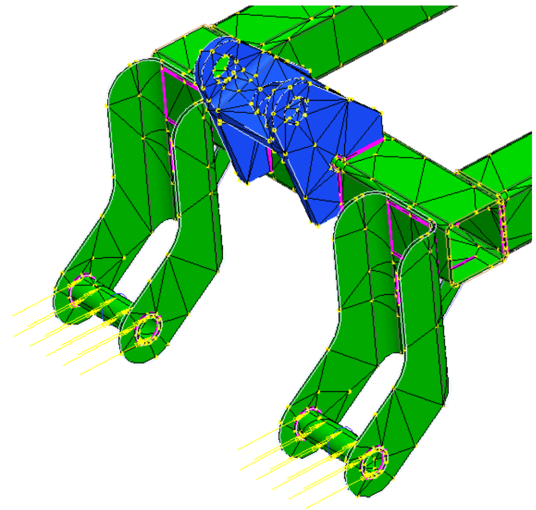
2. General Model Setup

General Remarks for modeling single parts
"Shell-friendly" features
Modeling joints with shells
The manual Mechanica weld types (see Mechanica Fundamentals II)
Automatic Mechanica weld types (see Mechanica Fundamentals II)
Important remarks for modeling assemblies
Assembly and part accuracies
Unit systems
Nomenclature
Final model checks
Archiving



3. Some Strategies for Building up Welded Assemblies with Shells

Simulation shell model as independent volume model
Example: Support frame as independent volume model
Simulation shell model as independent midsurface model
Example: Support frame as independent midsurface model
Simulation shell model as independent analysis assembly
Example: Support frame as independent assembly model
Simulation shell model as dependent part or subassembly in a design assembly:
-Using a volume-surface skeleton
-Using a midsurface skeleton



This workshop is not intended to show the one and only correct way to deal with huge shell assemblies. Instead, it supports you to develop your own methods while designing and modeling in Pro/ENGINEER, keeping in mind that you want to analyze your assembly with shells in Mechanica afterwards.